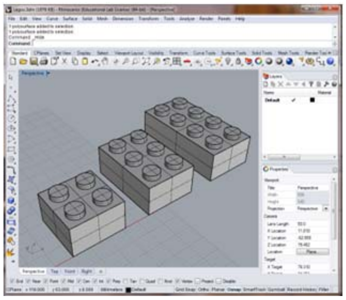
Preparing files for the 3d Printers with Rhino

Digital Media Tutorial
Written by Trevor Williams

What type of file should you prepare?

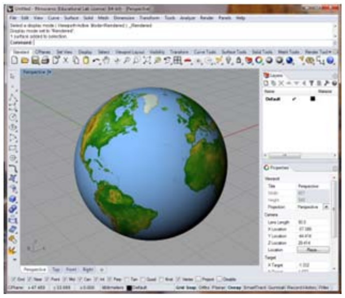
There are a myriad of filetypes used in 3d printing and rapid prototyping today. Within the School, and the majority of the professional world, there are only two filetypes that are used regularly. These are the STL (Stereolithography) and the VRML (Virtual Reality Modeling Language). STLs are files that record all the points and edges of a mesh and designate the inside and outside of faces. VRMLs record the same information regarding geometry, but also encode color, transparency, and shininess to the points of the mesh. The colors can be mapped either directly to the points (i.e. each point is also assigned an RGB set of values along with XYZ data) or they can be mapped via a texture-mapping process assigned in a modeler, where image files are wrapped around the object in a specified way.

In short, for a monochrome object, you should to prepare an STL file, and for a full-color object, you should prepare a VRML file.



```
1 polysurface added to selection
1 polysurface added to selection
Command: _Hide
Command:
```

These blocks can be printed in any single color, default being white. They should be exported as STLs.



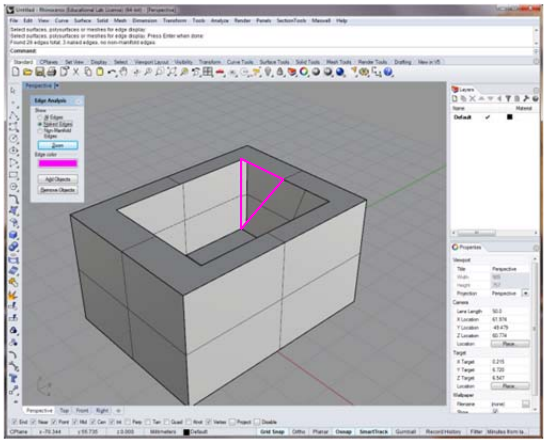
```
Select a display mode ( Viewport=Active Mode=Rendered ): _Rendered
Display mode set to "Rendered"
1 surface added to selection
Command:
```

This globe will be printed in full color. It should be exported as a VRML

Closing your object

There are many terms used for essentially the same thing in 3d printing. Watertight, manifold, closed, solid, and volumetric are all ways of describing a model has no holes in its surface and defines a clear interior volume from an exterior space. It is always advisable to close your object completely before sending the model to print, though in the case of VRMLs, it isn't always possible.

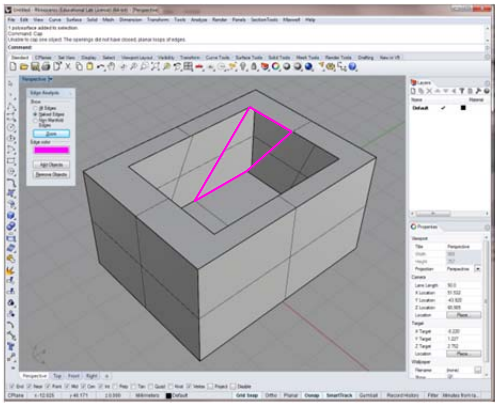
Most modeling will be done in NURBS surfaces, with meshes, and subdivision surfaces also being used, though less commonly. All these types of geometries can be analyzed for closed volumetricity in Rhino with the *ShowEdges* command.



```
Select surfaces, polysurfaces or meshes for edge display
Select surfaces, polysurfaces or meshes for edge display. Press Enter when done
Found 28 edges total; 3 naked edges, no non-manifold edges.
Command:
```

The ShowEdges command being run on this box shows the triangular opening inside the cavity in the box. The hole can be closed with the Cap command.

The easiest way to fill holes in a NURBS surface is through the command *Cap*. This will fill any closed, planar, holes in your surface geometries.



```
1 polysurface added to selection.
Command: Cap
Unable to cap one object. The openings did not have closed, planar loops of edges.
Command:
```

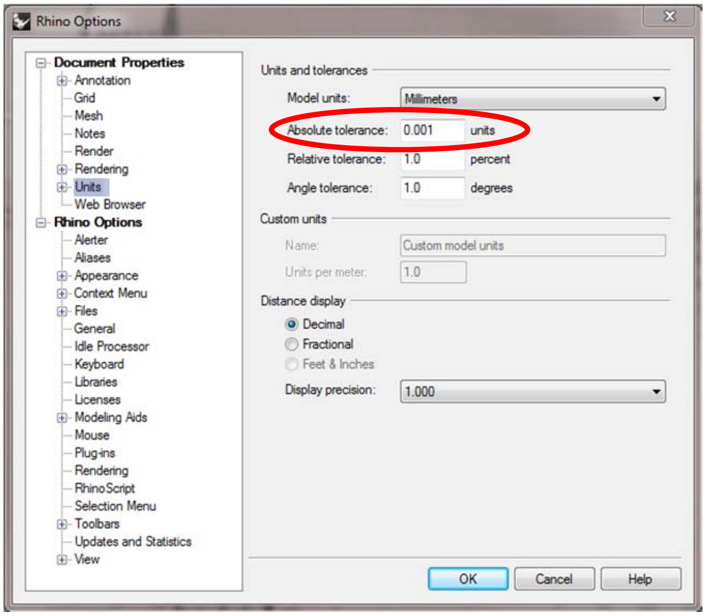
This hole is no longer planar, and must be closed with a different strategy.

For non-planar holes, different strategies will need to be employed. The commands *Patch*, *Loft*, *Sweep*, *Sweep2*, *EdgeSrf*, and *SrfPt* are all useful for generating NURBS surfaces that fit to the edges of holes. You should experiment with them to determine the best command to close your particular geometry.

Adjusting tolerances

When your polysurfaces are not closing easily, you may want to consider changing your model tolerances. These settings can be found by typing the *Options* command and going to the *Units* page.

The *Absolute Tolerance* is what determines the smallest unit that Rhino will consider when calculating limits of a surface or a curve. By default, Rhino will set your absolute tolerance to one ten-thousandth of your model's base unit. This is an extreme amount of precision that most modelers, especially architectural modelers, don't need. By setting the absolute tolerance higher (0.05 - 0.01 is usually sufficient) you can have Rhino fudge the gaps between two or more surfaces or curves and join them, even when they don't match perfectly on two edges or end-points.



Units and tolerances

Model units: Millimeters

Absolute tolerance: 0.001 units

Relative tolerance: 1.0 percent

Angle tolerance: 1.0 degrees

Custom units

Name: Custom model units

Units per meter: 1.0

Distance display

● Decimal

○ Fractional

○ Feet & Inches

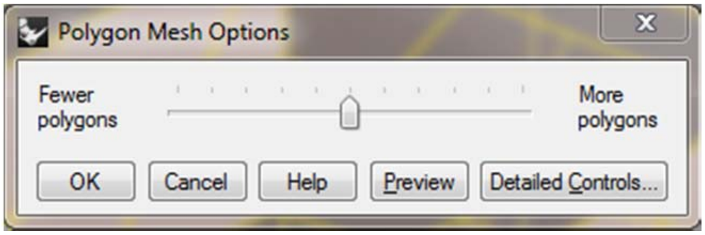
Display precision: 1.000

Most architectural scale models will still be significantly accurate at .01 units, whether that is millimeters or inches.

The benefit is that it will be much easier to join geometries. The downside is that a lot of modeling at a less precise tolerance may cause your model to eventually not fit exactly to your intended dimensions. Rhino will nudge or move curves and points within that tolerance in an attempt to make your geometries fit one another.

NOTE: To best mitigate that effect, do most of your modeling at a very precise (i.e. small) tolerance. Once the time comes to join together all your surfaces for a single uniform mesh, increase your tolerance, and then join your surfaces together.

Creating a mesh



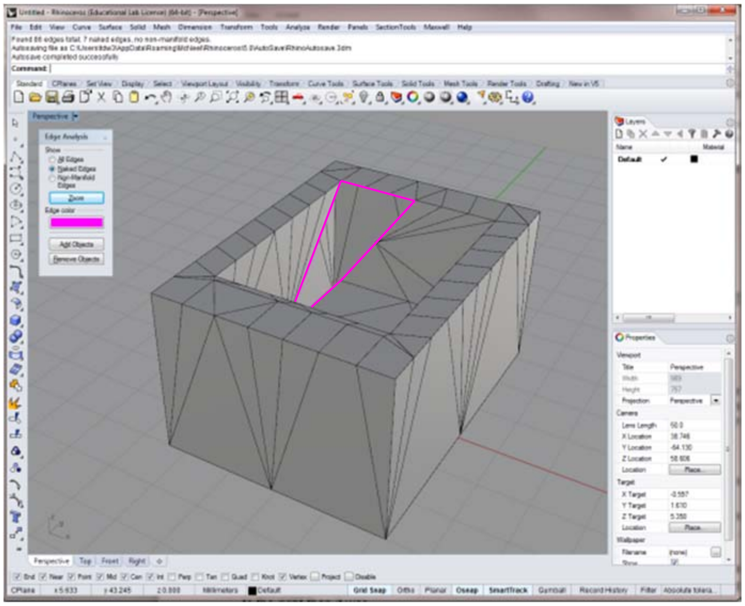
If you still cannot close your object while it is a NURBS object, you can attempt to do so after you've changed it to a mesh object using the *Mesh* command. When meshing, you want to create as simple (low-polygon count) of a mesh as possible. The slider that comes up should go to the right based on how many curved surfaces you have and the size of your intended object. Follow these rules of thumb:

- If your object is entirely rectilinear, drag the slider all the way to the left.
- If you have a large object (greater than 5 in^3 in bound volume) with some softly curved surfaces, move the slider 10-20% to the right.
- If you have a large object with sharply curved surfaces, move the slider to 40-60% to the right.
- If you have a small object (less than 5 in^3 in bound volume) with softly curved surfaces, move the slider 50-75% to the right.
- If you have a small object, with sharply curved surfaces, move the slider 80-100% to the right.
- If any important details on your object have disappeared or become misshapen, move the slider further to the right than it was in 5-10% increments until an acceptable level of detail is reached.

Closing mesh holes

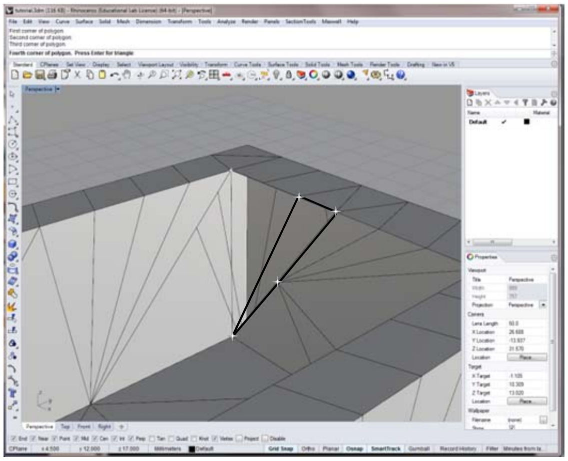
With a mesh, you are no longer working with any truly curved surfaces, or any surfaces at all. Your object will be changed into flat triangles and quadrangles called faces. These are composed of edges and vertices, which are what you will manipulate when working with a mesh.

The *ShowEdges* command still is used to find and focus on any holes or openings in your mesh. If you have only a few holes, you may just want to use the *FillMeshHole* command. If you have many holes, or are having trouble finding holes, you may be better served by *FillMeshHoles*, which will automatically attempt to close all naked edges. These commands draw new mesh faces between the edges of holes in your mesh.



```
Found 86 edges total; 7 naked edges, no non-manifold edges.
Autosaving file as C:\Users\tdw3\AppData\Roaming\McNeel\Rhinoceros\5.0\AutoSave\RhinoAutosave.3dm
Autosave completed successfully
Command:
```

The mesh version of the box from earlier still has the same non-planar hole.



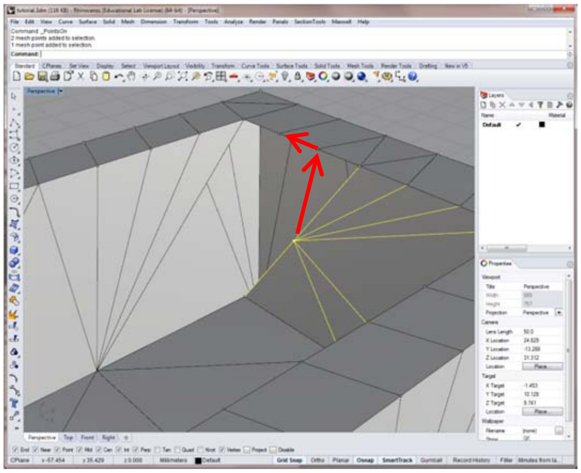
```
First corner of polygon.
Second corner of polygon.
Third corner of polygon.
Fourth corner of polygon. Press Enter for triangle
```

When using 3dFace it's important to draw a vertex anywhere there is one along an open edge. Even if two edges run in a straight line, but are broken by a vertex, you should still place a vertex at the same point as the others or your edges will not close.

If using those commands still leaves holes in your mesh or create unwanted faces, you can fill your holes by hand by using the *3dFace* command and selecting 3 or 4 points at a time, which will define a new mesh face. When you have closed your mesh with new faces, you can make them into a single, closed mesh with the *Join* command.

If the open edges of one or more meshes are very close, but do not meet evenly, or have different numbers of vertices, you can select the mesh vertices and either delete them or move them to match up with one another. Do this by selecting the meshes you are trying to close, then hit *F10* to pull up the control points interface. Then, making sure that *Vertex* is selected in your Snap Menu, you can proceed to close the mesh.

NOTE: Any meshes, whether they share vertices or not, can be made into one mesh with the *Join* command. However, a mesh will only be closed by *Join* if the input meshes share edges and vertices. If you join meshes that do not shares edges and vertices you will essentially just group them.

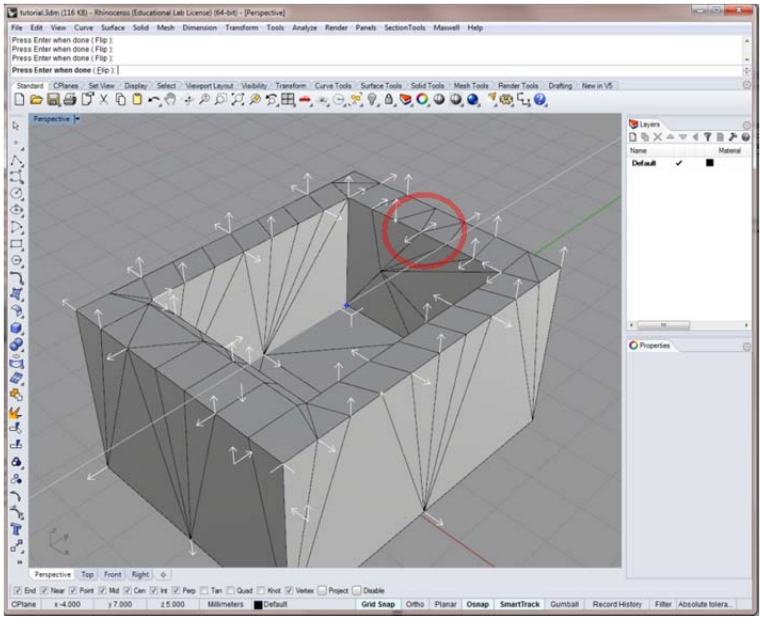


```
Command: _PointsOn
2 mesh points added to selection.
1 mesh point added to selection.
Command:
```

After turning on the control points for this mesh, I can close it partially by dragging the selected vertices to the next vertex along the open edge. I could close it further by dragging all those vertices to the next vertex on the open edge.

Normalizing mesh faces

When your mesh has been closed, it's possible that not all of the mesh normals (the directions that designate inside and outside for each mesh face) are uniformly aligned. This means that your object, while closed, may not be clearly identifying the interior and exterior volumes for your object. This can lead to a number of problems with your final print, particularly by adding extra faces and blocking off portions of your model by the printer.



```
Press Enter when done ( Flip ):
Press Enter when done ( Flip ):
Press Enter when done ( Flip ):
Press Enter when done ( Flip ):
```

☑ End ☑ Near ☑ Point ☑ Mid ☑ Cen ☑ Int ☑ Perp □ Tan □ Quad □ Knot ☑ Vertex □ Project □ Disable

CPlane x -4.000 y 7.000 z 5.000 Millimeters Default

tutorial.3dm (116 KB) - Rhinoceros (Educational Lab License) (64-bit) - [Perspective]
File Edit View Curve Surface Solid Mesh Dimension Transform Tools Analyze Render Panels SectionTools Maxwell Help
Standard CPlanes Set View Display Select Viewport Layout Visibility Transform Curve Tools Surface Tools Solid Tools Mesh Tools Render Tools Drafting New in V5
Perspective
Layers
Name
Material
Default
Properties
Perspective Top Front Right
Grid Snap Ortho Planar Osnap SmartTrack Gumball Record History Filter Absolute tolera...

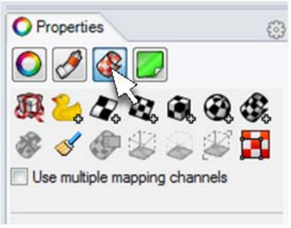
Look for arrows going in opposite directions, like in this image. They are a sign of incongruent surface normals.

It's possible to check the normal directions of all the faces of your mesh or the surfaces in a polysurface by selecting the object and using the command *Dir*. The arrows that appear show you the direction of the normal for every face and surface at their corners. If you see two arrows that go in opposite directions, that is a definite sign that you have two faces with normals that go in opposite directions. To ensure that all the normals of a mesh or polysurface go in one direction, use the command *UnifyNormals.* If the normals unify, but all point inside rather than outside use the *Flip* command to make them point outside, which is necessary for 3d printing.

Once the mesh is closed, you can export it as an STL and open it with any of the 3d printers' software packages.

Working with color on a mesh

There are two ways to produce a colored mesh (VRML) for 3d printing. One method, *texture mapping*, projects an image around the outside of a closed object. The other, *material mapping*, works by grabbing the rendered material color for each vertex of a mesh and correlating that to each vertex.

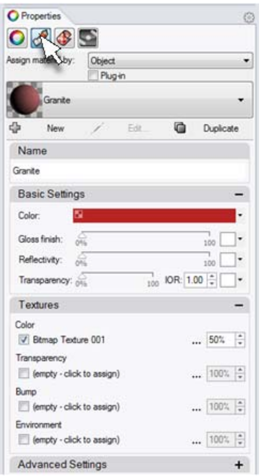


□ Use multiple mapping channels

If mapping a texture, you must select a method of map projection to use.

When mapping a texture in Rhino, you must specify a method of map projection. To do this, first assign a material or texture map. Then select the object, go to the properties tab, and select the *Texture Mapping* section. This is the third icon from the left. From there, you can select *Surface*, *Planar*, *Box*, *Spherical*, or *Cylindrical* mapping methods.

When mapping a texture in Rhino, you must specify a method of map projection. To do this, first assign a material or texture map. Then select the object, go to the properties tab, and select the *Texture Mapping* section. This is the third icon from the left. From there, you can select *Surface*, *Planar*, *Box*, *Spherical*, or *Cylindrical* mapping methods.



Properties

Assign material by: Object

□ Plug-in

Granite

New Edit... Duplicate

Name: Granite

Basic Settings

Color:

Gloss finish: 0% 100

Reflectivity: 0% 100

Transparency: 0% 100 IOR: 1.00

Textures

Color
☑ Bitmap Texture 001 ... 50%

Transparency
□ (empty - click to assign) ... 100%

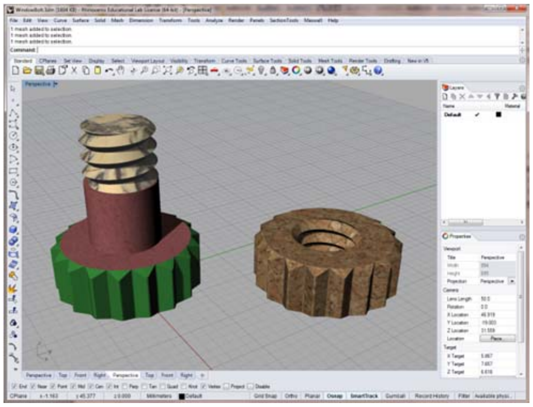
Bump
□ (empty - click to assign) ... 100%

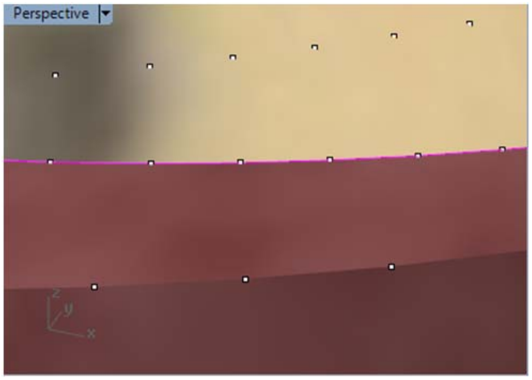
Environment
□ (empty - click to assign) ... 100%

Advanced Settings +

You can also use the regular material assignments in Rhino to map colors to your objects. For this method, you won't necessarily close all your objects. Instead you will need to have complementary (matching) mesh edges and vertices, though each set of faces with different materials will be their own separate mesh in the VRML file.

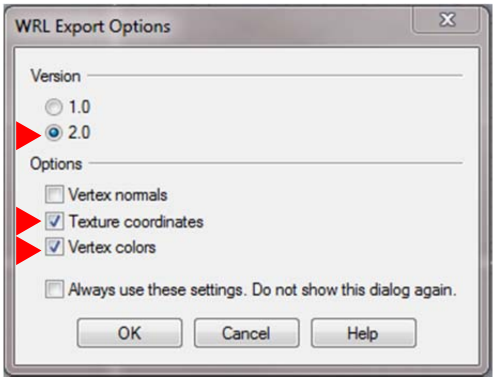
In the example on the right, the right object is a closed mesh. The leftt object is 3 different meshes with different material assignments that have complementary edges and vertices where the meshes meet. The highlighted naked edges and points match up exactly on both meshes. As a result, the two separate meshes will still be printed as one closed object, but both will also have their independent color information.





Exporting a mesh with color

When exporting from Rhino as a colored VRML, you should always export vertex colors, as this is the color information on each point. If using a texture map, it's also important to export the texture coordinates, as this will map the source image the same way in the VRML file as it is in your Rhino model. If you use a texture map, make sure the source image file is in the same folder as your VRML file when opening it in ZPrint. If you don't, ZPrint will ask to browse your computer for the file each time you open the model.



WRL Export Options

Version

○ 1.0

● 2.0

Options

□ Vertex normals

☑ Texture coordinates

☑ Vertex colors

□ Always use these settings. Do not show this dialog again.

These are the necessary Rhino export settings for a color model.